NPS-ME-97-005

NAVAL POSTGRADUATE SCHOOL
Monterey, California

Modal Analysis of the 72 Inch TAC-4 Ruggedized Rack (CLIN 0003AA)

by

Mark H. Oesterreich
Y. S. Shin

September 1997

Approved for public release; distribution is unlimited.

Prepared for: Naval Command, Control and Ocean Surveillance Center
Research, Development, Testing and Evaluation Division
San Diego, CA 92152-5000

Naval Postgraduate School
Monterey, California

Rear Admiral M.J. Evans
Superintendent

R.S. Elster
Provost

This report was prepared in conjunction with research conducted for Naval Command, Control and Ocean Surveillance Center, Research Development Testing and Evaluation Division.

This report was prepared by:

Mark H. Oesterreich
LT USN

Y.S. Shin
Professor of Mechanical Engineering

Reviewed by:

T.R. McNelley
Chairman
Dept. of Mechanical Engineering

Released by:

D.W. Netzer
Dean of Research

REPORT DOCUMENTATION PAGE

Form Approved OMB No 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE September 1997.

3. REPORT TYPE AND DATES COVERED FY 1997

4. TITLE AND SUBTITLE: Modal Analysis of the 72 Inch TAC-4 Ruggedized Rack (CLIN 0003AA)

5. FUNDING NUMBERS N6600197WR00121

6. AUTHOR(S) M.H. Oesterreich, Y.S. Shin

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Naval Postgraduate School
Monterey CA 93943-5000

8. PERFORMING ORGANIZATION REPORT NUMBER
NPS-ME-97-005

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
NCCOSC, RDT&E DIV SAN DIEGO, CA 92152-5000

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES The views expressed here are those of the authors and do not reflect the official policy or position of the Department of Defense or the U.S. Government.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(maximum 200 words)*

The finite element modeling and subsequent static and modal analyses of the 72 Inch TAC-4 Rugged Rack computer system (CLIN 0003AA), as designed by Science Applications International Corporation (SAIC), has been performed to determine the natural frequencies and corresponding mode shapes for subsequent transient shock analysis. This rack is designed to allow incorporation of commercial-off-the -shelf (COTS) computer systems for naval tactical computing requirements while still meeting applicable shock and vibration mil-specs.

14. SUBJECT TERMS TAC-4, modal analysis

15. NUMBER OF PAGES 40

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	SAR

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. 239-18 298-102

ABSTRACT

The finite element modeling and subsequent static and modal analyses of the 72 Inch TAC-4 Rugged Rack computer system (CLIN 0003AA), as designed by Science Applications International Corporation (SAIC), has been performed to determine the natural frequencies and corresponding mode shapes for subsequent transient shock analysis. This rack is designed to allow incorporation of commercial-off-the -shelf (COTS) computer systems for naval tactical computing requirements while still meeting applicable shock and vibration mil-specs.

TABLE OF CONTENTS

LIST OF FIGURES

1. INTRODUCTION

A. OVERVIEW

The rapid pace of current electronic and computer modernization coupled with the slow turnaround and life-cycle outlook of traditional naval contracting has precipitated a situation where the navy can not keep pace with current technology. As a further difficulty in the contracting process, all military-specifications (mil-specs) must be met by the manufacturer. This adds significant cost and design and manufacturing delays to any computer system required for naval tactical use. These design, manufacture, and contracting delays result in a product which is obsolete well before it can be placed into service.

To remedy this situation, the U.S. Navy has changed its contracting procedures, but more importantly the type of computer hardware used for tactical computing. Previously, each tactical computer was a stand-alone unit specifically designed for its requisite task, or relied on a standard chassis which was modified to perform the required task (e.g. AN-UYK-7,-43). This approach was satisfactory until the advent of the computer revolution which heralds a new technical innovation or increase in computing capacity about every six months. This time period is too short for the industry to design mil-spec computers at the same rate as commercial computers improve. The Navy is now implementing the use of COTS computers in tactical applications.

The benefits of this introduction of this commercially based strategy will reduce delays in introducing new technology to the fleet, reduce software development and logistical costs, and improve long-term compatibility and reusability of tactical information technology investments. However, the survivability of COTS in various types of severe environments is questionable. The TAC family uses commercial computers and places them into special ruggedized racks which are designed to meet all applicable mil-specs thereby allowing the use of COTS equipment in a tactical environment.

B. RESEARCH OBJECTIVES

There are three different sizes of ruggedized racks used in the TAC-4 system, each size rack also has varying equipment types and configurations. In this work, the analysis is focused on the 72 inch TAC-4 rack with the CLIN 003AA configuration which has already begun testing for its compliance with the shock and vibration mil-specs (MIL-STD- 901D for shock and MIL-STD-167-1 & MIL-STD-810, Method 514, Procedure 1, Category 8 for vibration). Due to the nature of the use of this computer system, the rack must reduce all specified shocks and vibrations to a level that the non-hardened commercial computer hardware can handle without system failure (interruption of service).

The purpose of this study is to build a finite element model of CLIN 0003AA as developed by SAIC [Ref. 1] and perform static and modal analysis of the model. This study will lead into subsequent studies of the finite element model's shock and vibration response for comparison with actual test data. Once the computer model is perfected and verified, it can be used to study various rack and rack configuration changes without the need for extensive actual physical testing of each rack type.

2. FINITE ELEMENT METHOD

The finite element method is a numerical procedure for approximating the solution to many complex problems encountered in engineering. An approximate solution to the theoretical behavior of complex structures are obtained at finite points of the model. The accuracy of a finite element solution depends on the way the finite element model is generated. That is, it depends on the number and type of elements used in the model.

Free vibration analysis (Modal Analysis) is a method for predicting the undamped vibration characteristics of a structure. This vibration takes place in the absence of external excitation. The equation of motion of a discrete system is written in matrix form as,

$$[M]\{\ddot{q}\} + [K]\{q\} = \{0\} \tag{1}$$

where, [M] is the structural mass matrix, $\{\ddot{q}\}$ is the nodal acceleration vector, [K] is the structural stiffness matrix and $\{q\}$ is the nodal displacement vector. Equation (1) is a system of coupled differential equation with n-independent unknowns, where n is the total number of degrees-of-freedom in the structure. By assuming a solution of the form,

$$\{q\} = C\{\varphi\}e^{j\omega t} \tag{2}$$

an n^{th} order homogenous eigenvalue problem is generated as,

$$\left[-\omega^2[M] + [K]\right]Ce^{j\omega t} = 0 \tag{3}$$

where, C is constant and $\{\varphi\}$ is a spatial vector. The non-trivial solution to equation (3) is,

$$\det\left[-\omega^2[M] + [K]\right] = 0 \tag{4}$$

This is the characteristic equation of the system. The roots of this equation are the eigenvalues, $\lambda_i = \omega_i^2$, where each ω_i is the natural frequency of vibration. For each natural frequency, there is a corresponding eigenvector $\{\varphi\}$ that is the mode shape associated with that frequency. Thus, the solution to the free vibration is n eigenpairs, ω_n and $\{\varphi^n\}$.

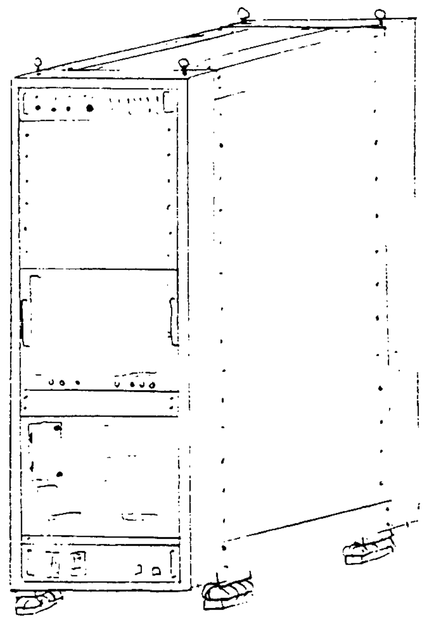

Figure 1. 72 inch TAC-4 Ruggedized Rack (CLIN 0003AA)

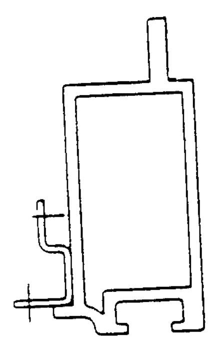

Figure 2. Cross Section of Actual Tubular Frame

3. FINITE ELEMENT MODELING

The finite element (FE) model of the TAC-4 CLIN 0003AA was built using MSC/PATRAN 6 code. The first step in finite element analysis was to build a solid model using the drawings provided in Reference (2). Using the solid model as a geometric reference, a finite element model was built using manually defined nodes and elements. Certain features of the solid model such as bolt holes, electrical connections, and fasteners were not transferred to the model in order to not introduce large numbers of elements. Also, certain complicated geometric features of the model, such as drawer slides and flanges, were simplified for the same reason.. Once a complete finite element model of the rack was generated, physical and material properties of the various parts were then defined and a boundary condition was applied to the model. Static analysis of the model was used to verify that it behaved appropriately to check that there were no gross defects in the model. The modal analysis was carried out using MSC/NASTRAN [Ref. 3] code.

A. SOLID MODELING

This section provides a description of the modeling of the various portions of the 72in rack for this and subsequent analyses. Figure 1 is a picture of the rack. The rugged rack consists of a metal tubular frame with metal cover plates enclosing a metal drawer frame which houses the electronic equipment drawers in a sliding rail arrangement. Due to the complicated internal structure of the rack, for initial modeling purposes, it was broken down into several manageable sections and then recombined to form the entire structure.

The following paragraphs describe the modeling of each individual portion of the rack.. The rack was broken down into the following sections and will be referred to with the following names: tubular frame, rack frame, shell, and drawers. With the exception of the drawers, all parts of the rack system were modeled as 18-8 stainless steel of the appropriate thickness. From the outset, the goal of the modeling process was to minimize the number of elements in order to minimize the computational requirements, but to still ensure an accurate model response. A large portion of this strategy is to

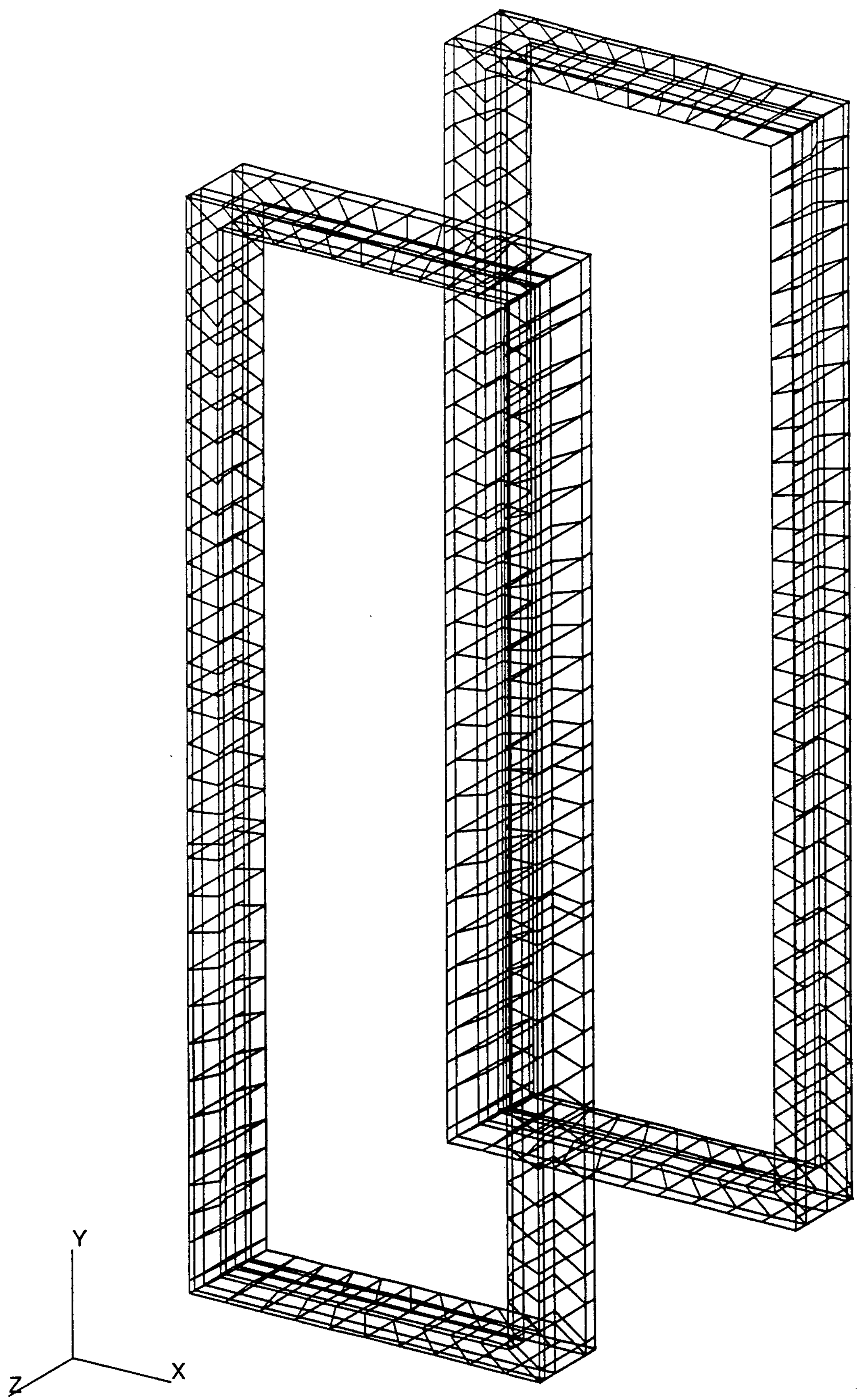

Figure 3. Finite Element Model of Tubular Frame

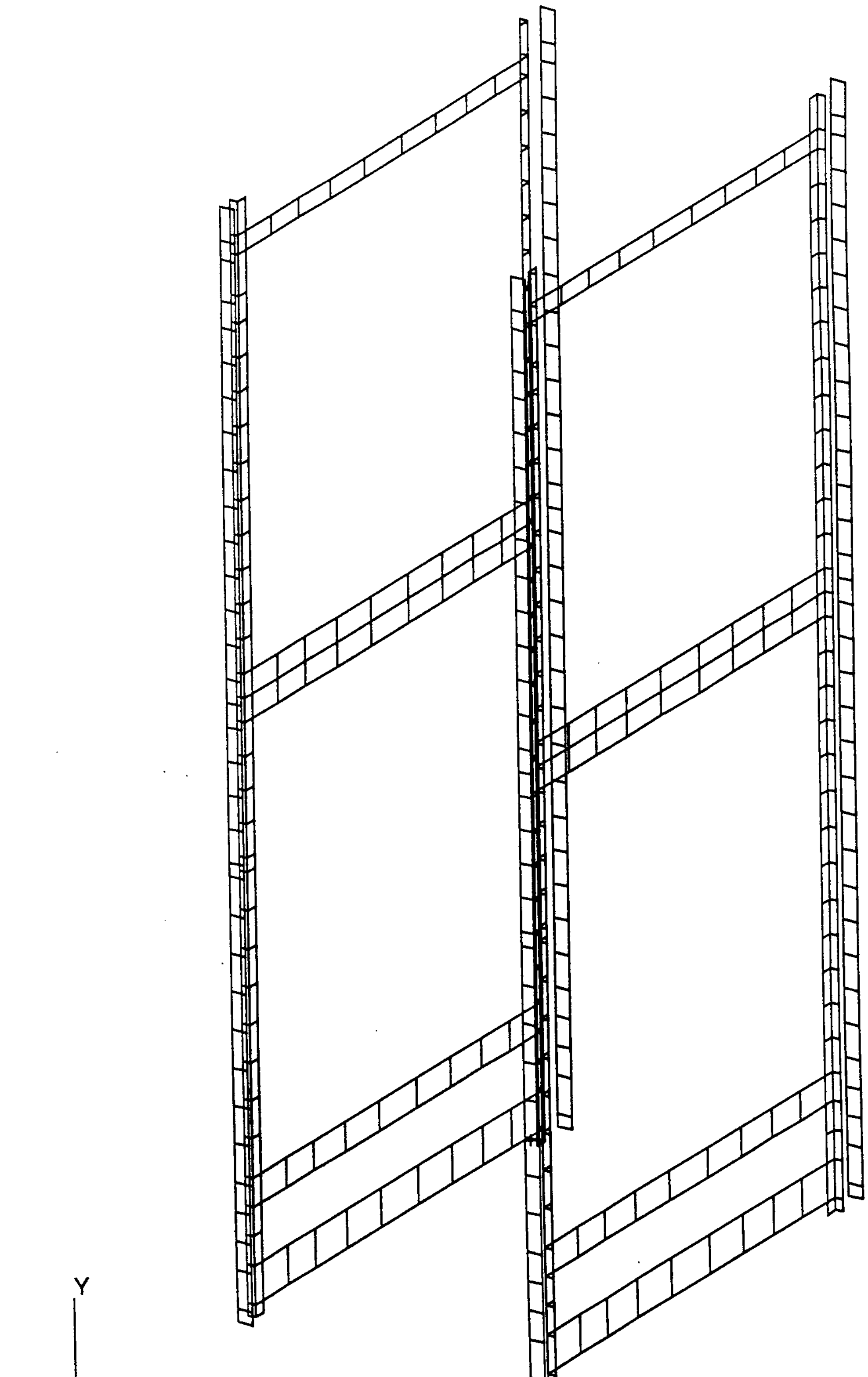

Figure 4. Finite Element Model of Rack Frame

simplify the actual structure shapes into idealized structures. The next most important part of this strategy, which goes hand in hand with the first, is to make the individual elements as large as possible. These two concepts are the underlying reason for all modeling decisions - to keep the mathematical model as small as possible with minimal impact on solution accuracy.

The tubular frame is the portion of the rack which is the gross load bearing structure in the rack. In the actual rack, this has a complicated shaped structure as shown in Figure 2. Because the various channels and mini-flanges on this structure would severely complicate the modeling of it, this structure was simplified to a rectangular box of similar dimensions. This will provide for a similar dynamic model response. The large flanges on the tubing, however, were not ignored because of their comparatively large dimensions. Figure 3 is a diagram of the finite element model of this portion of the rack system.

The rack frame is the portion of the rack which the drawers are connected to using sliding rail mounts. Due to the fact that the large numbers of small bolt holes in this portion of the model would severely complicate the model, all bolt holes were ignored. Although holes in any solid material significantly affect that piece's structural properties (e.g. bending, torsion), the effects can easily be compensated for by changing the materials other properties (Young's Modulus, thickness) to account for this. This portion of the model also includes the drawers' sliding rail mounts. These pieces are a very complicated nesting of steel channels, rollers, and cam locks. Since this would be very difficult to model, these pieces were modeled as solid metal bars. Again, material properties can easily be varied to account for the actual behavior of these rails. Figure 4 is the finite element model of this portion of the rack system.

The shell consists of all the cover plates in the rack system: top, bottom, sides, front, and back. These were very simple to model because they are simply flat plates. Figure 5 is the finite element model of this portion of the rack system.

The drawers due to their complicated internal structure also had to be simplified in the modeling process. The drawers were modeled as solid blocks of the appropriate size with a corresponding density so they had the correct weight. Other materials were

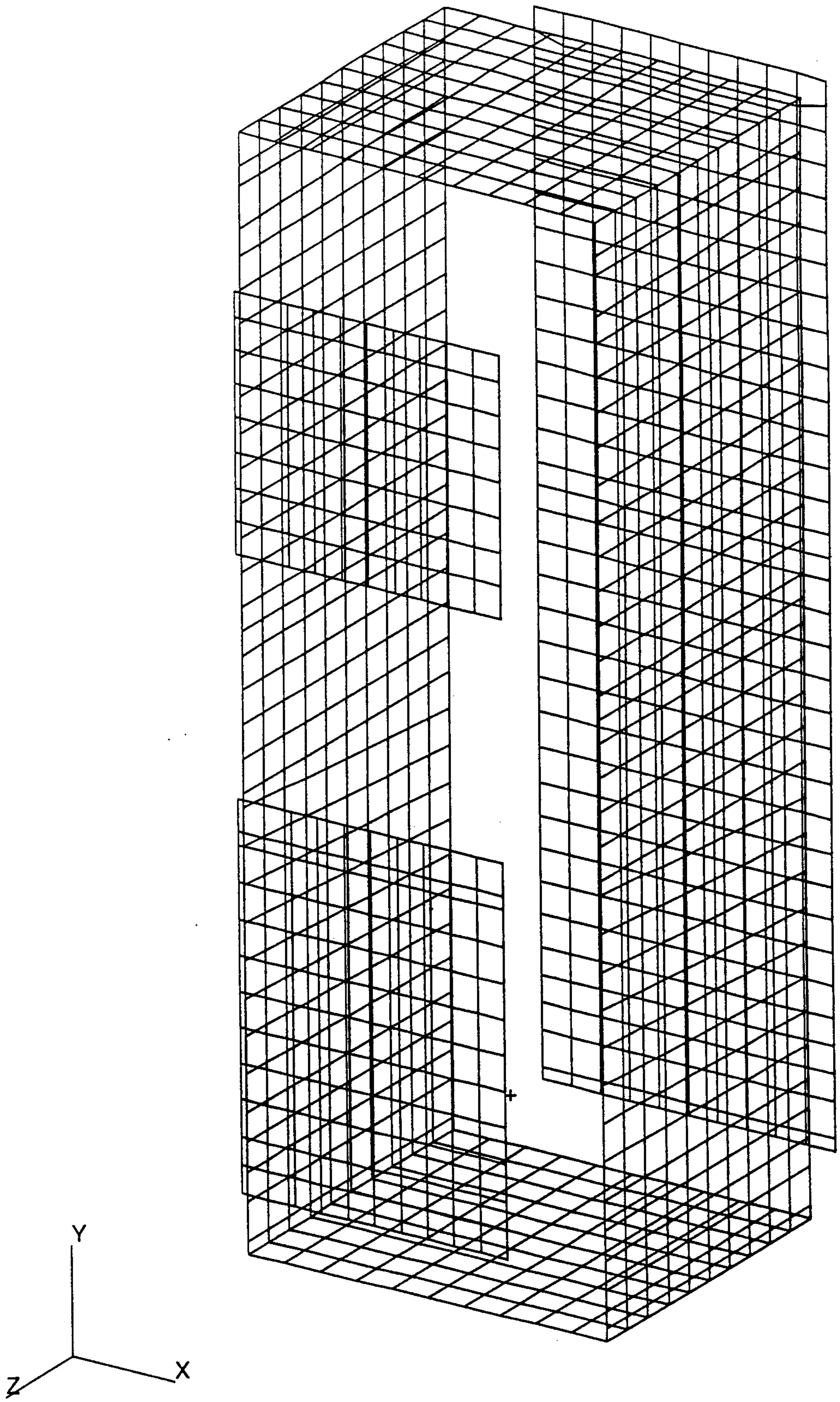

Figure 5 Finite Element Model of Shell (Cover Plates)

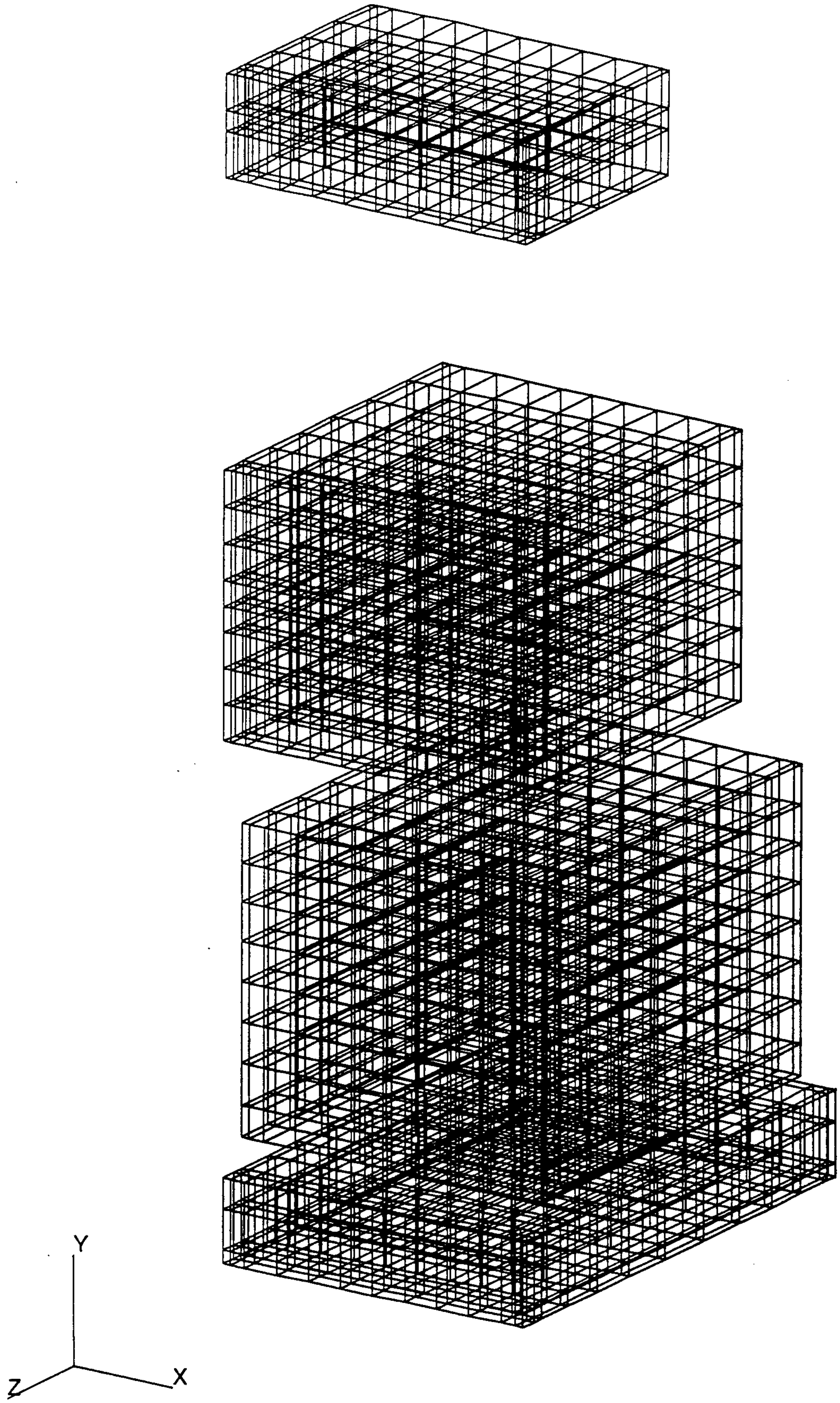

Figure 6 Finite Element Model of Drawers

dynamic response. Figure 6 is the finite element model of this portion of the rack system.

Once each portion of the rack system was discretized, they were appended to each other to form the entire rack system assembly. Once this was complete, all possible model verification checks were performed. All elements were checked to ensure that their dimensions and shapes were well within acceptable standards to ensure the accuracy of all ensuing model calculations. Also, checks for duplicate nodes and elements, missing nodes and elements were performed. If the model is properly constructed, the free edges will show the actual model boundaries. If not, a free edge will appear to indicate either a crack or a hole in the model.

B. MODEL EVOLUTION

From the outset, the thin shell finite element was used for all metal components in the rack system. This was done simply because of the fact that this type of element mathematically models the dynamic behavior of flat, thin items much more accurately than solid elements which have difficulties in modeling twisting which most certainly will occur in the actual dynamic response of the rack system.

Due to the complication of the internals of the drawers, the first model of this system sought to model the drawers as point masses connected by rigid bars to the frame rails. Difficulty arose with this model, however, once the modal analysis of it was completed and the data analyzed. It was readily apparent that the modeling of the drawers as point masses would not provide an accurate model response in the vicinity of the drawer rails. By modeling the drawers as point masses, the mountings to the drawers became point loads distributed to only two nodes of each rail. Since in reality, the drawers provide a distributed load to the rails, the loads needed to be distributed to more nodes on the respective rails. Also the point masses themselves did not accurately simulate the actual motion the drawer would encounter, making the future transient acceleration responses completely inaccurate.

At this point the drawers were remodeled as the solids described in the previous section. Once this remodeling was complete all of the same model checks performed on the previous model were completed. At this point it also must be made clear that both of these finite element model do not include the shock isolators. This was done primarily to simplify the initial model verification calculations - essentially the primary goal was to obtain, as rapidly as possible, an excellent basic model from which to work from.

4. STATIC ANALYSIS

All static analysis of the model were performed using MSC/NASTRAN finite element structural code. A static analysis was performed on the model as a gross model integrity check, that is to see if any unknown seams or holes appear that were not previously detected. The other reason for this analysis is to accomplish a final common sense check of the model prior to performing the more elaborate dynamic calculations. This analysis also serves as a quick check of the modeling of each individual component. This is accomplished by comparing the calculated static deflection of a component with normal gravity applied, for example - the center of a drawer rail, with what you would expect for the given loading condition. MSC/PATRAN 6 contains a utility which will display graphically using color shading the calculated results of the ranges of deflection for a given loading case. This allowed a quick check to see what the largest deflection was and where it was. All values for deflections in the model were on the order of 100ths to 1000ths of an inch which were within the actual expected values.

5. NORMAL MODE ANALYSIS

The normal modes of vibration of the model were solved using MSC/NASTRAN finite element structural code. The solutions are the system undamped natural frequencies and their mode shapes as discussed in the section on the Finite Element Method.

All of the mode shapes below 100 Hz were obtained and are tabulated in Appendix A with a description of the motion. Due to the connectivity of each part in the finite element model, resonant frequencies of one component has an effect on the other components to some degree. In the following discussion the mode shapes are depicted by

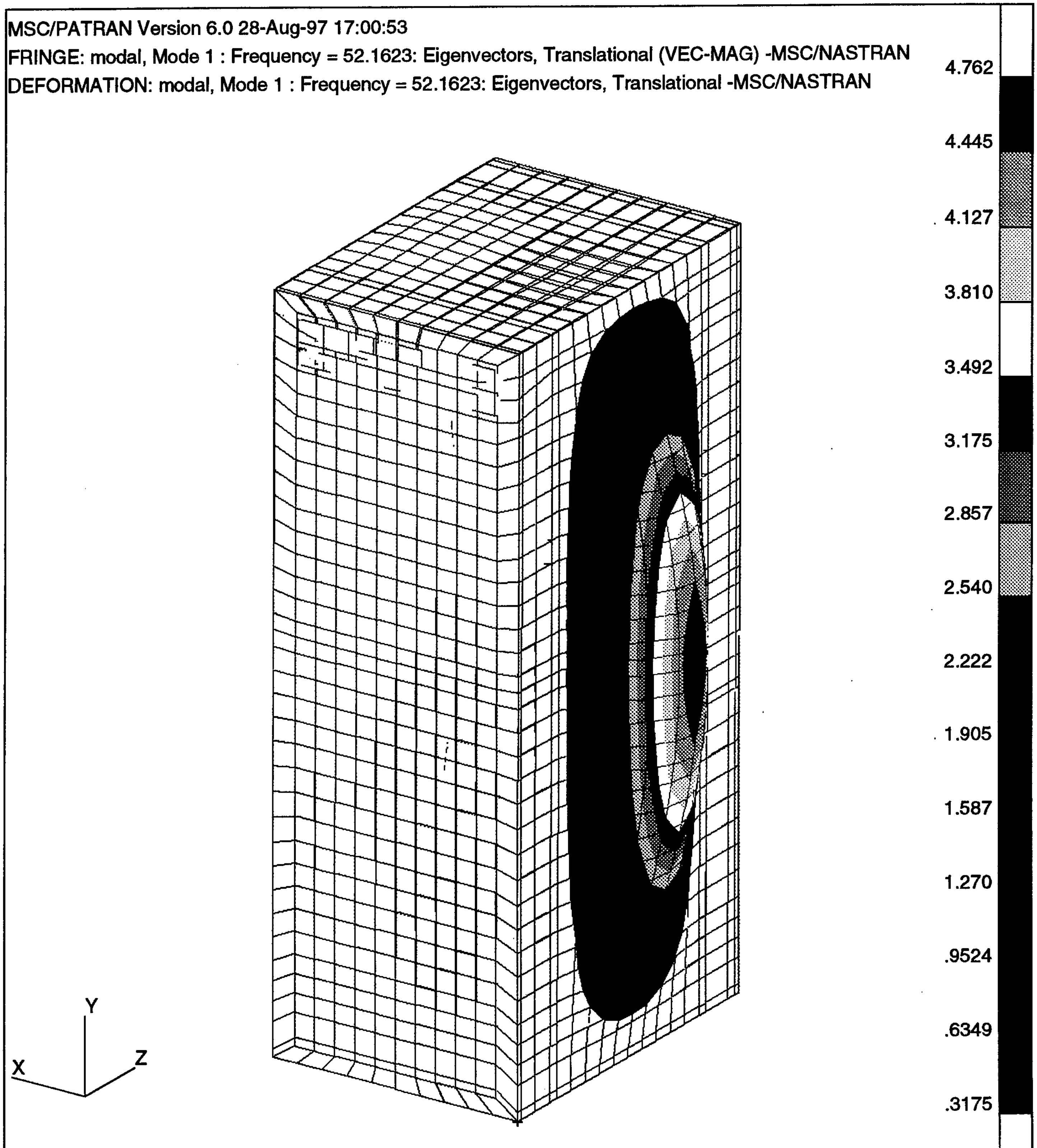

Figure 7. Mode 1, 52.16 Hz, Side Local Plate Bending Mode

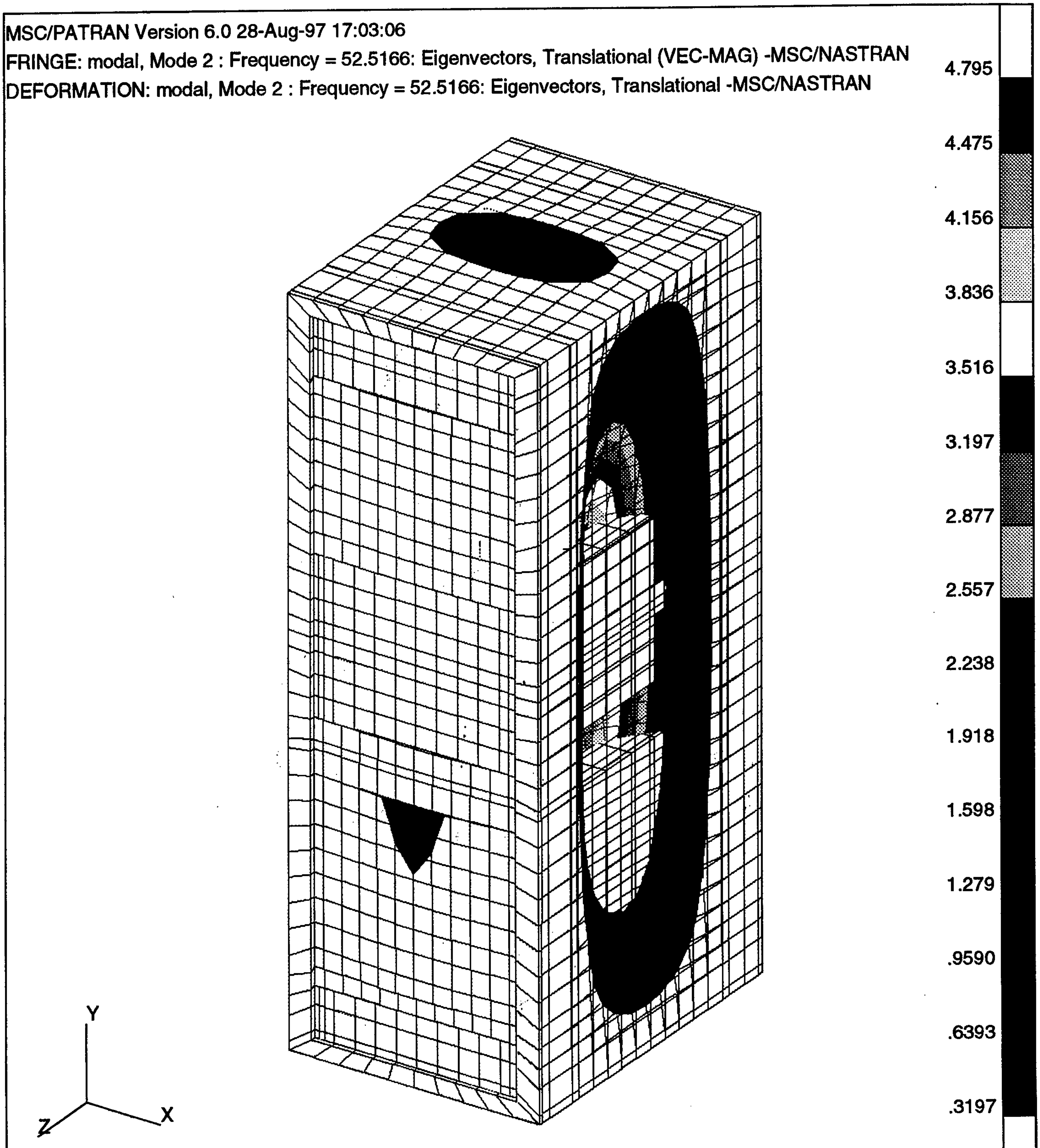

Figure 8. Mode 2, 52.52 Hz, Side/Top Local Plate Bending Modes

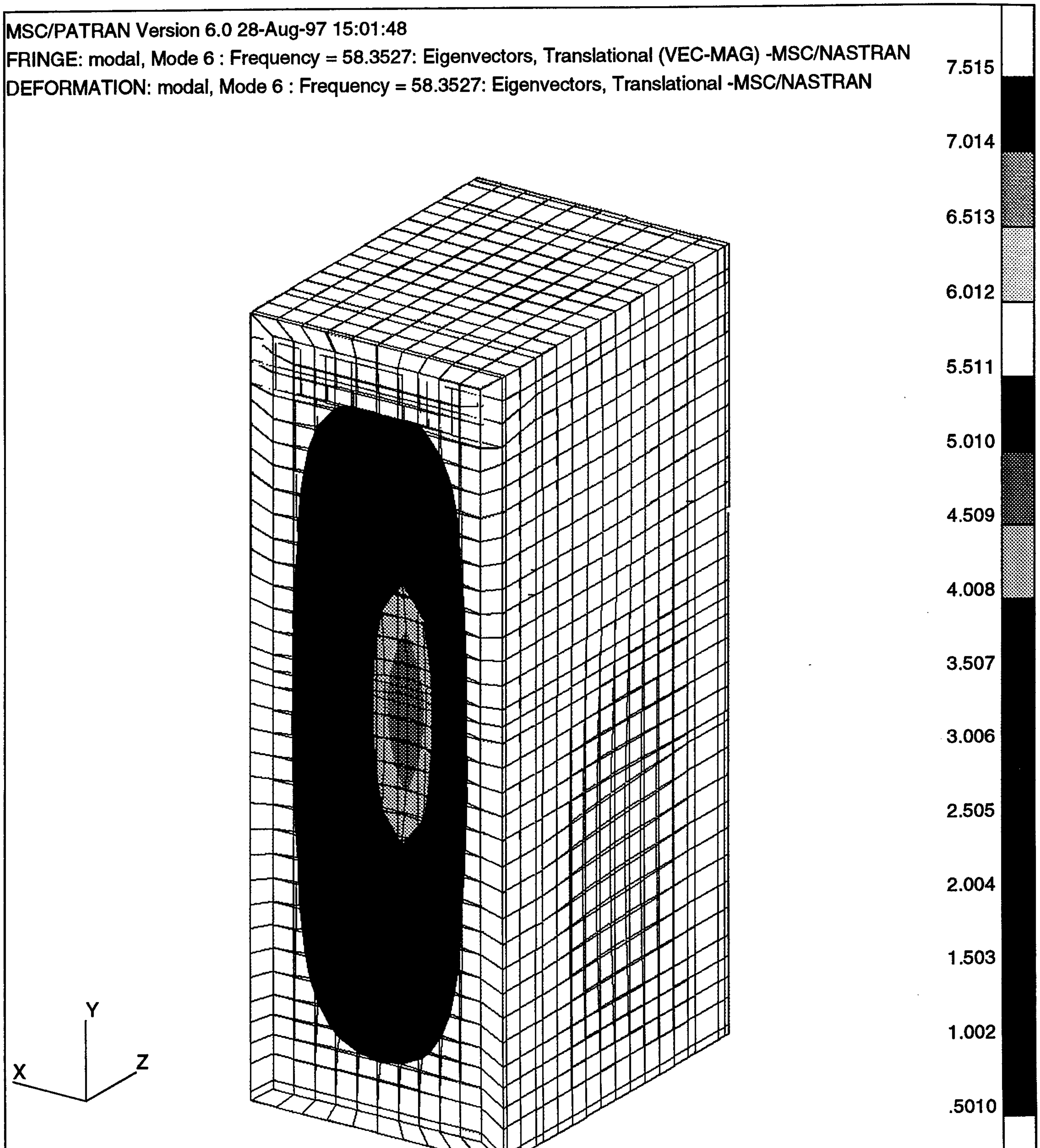

Figure 9. Mode 6, 58.35 Hz, Rear Panel Local Bending Mode

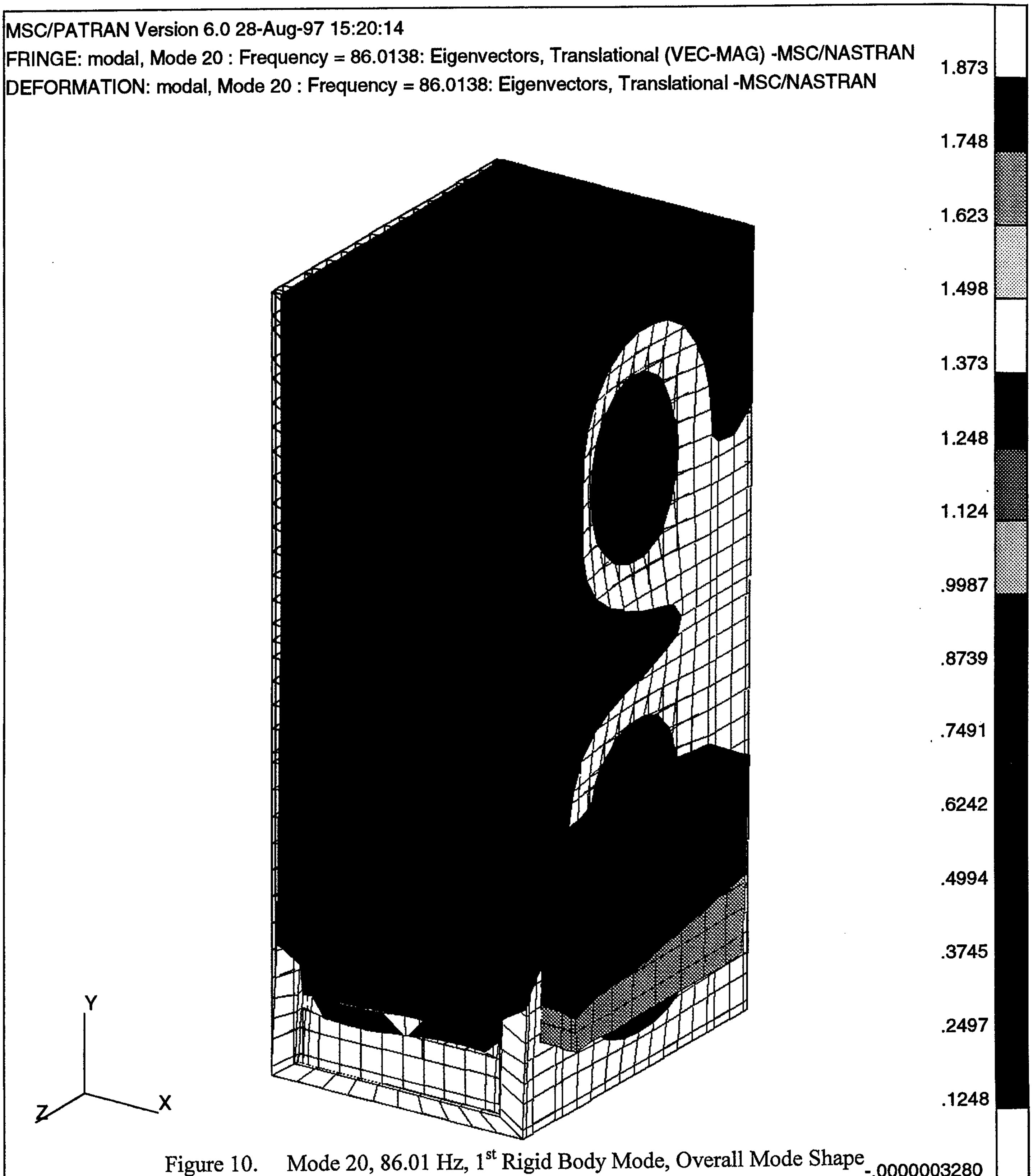

Figure 10. Mode 20, 86.01 Hz, 1st Rigid Body Mode, Overall Mode Shape

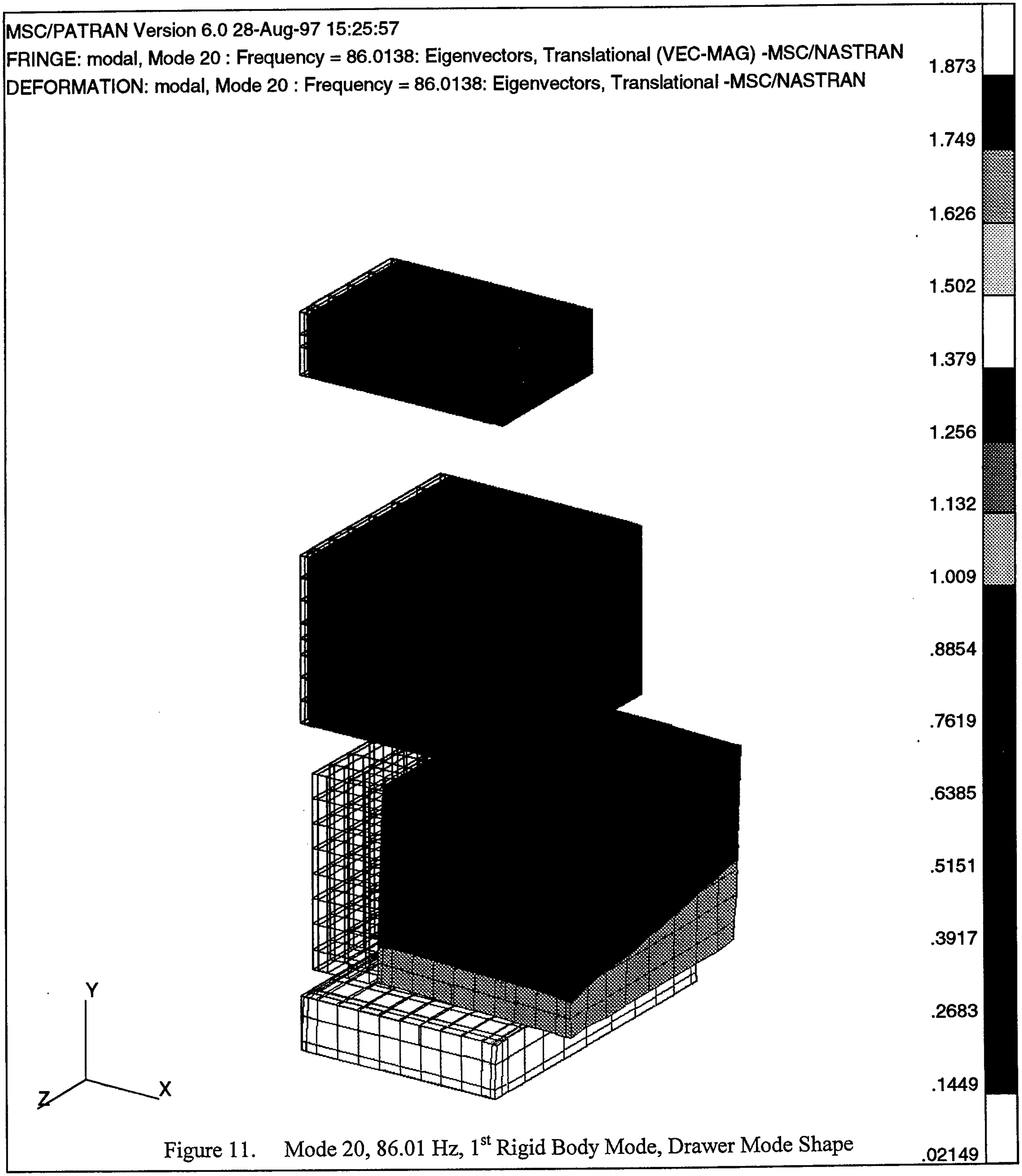

Figure 11. Mode 20, 86.01 Hz, 1st Rigid Body Mode, Drawer Mode Shape

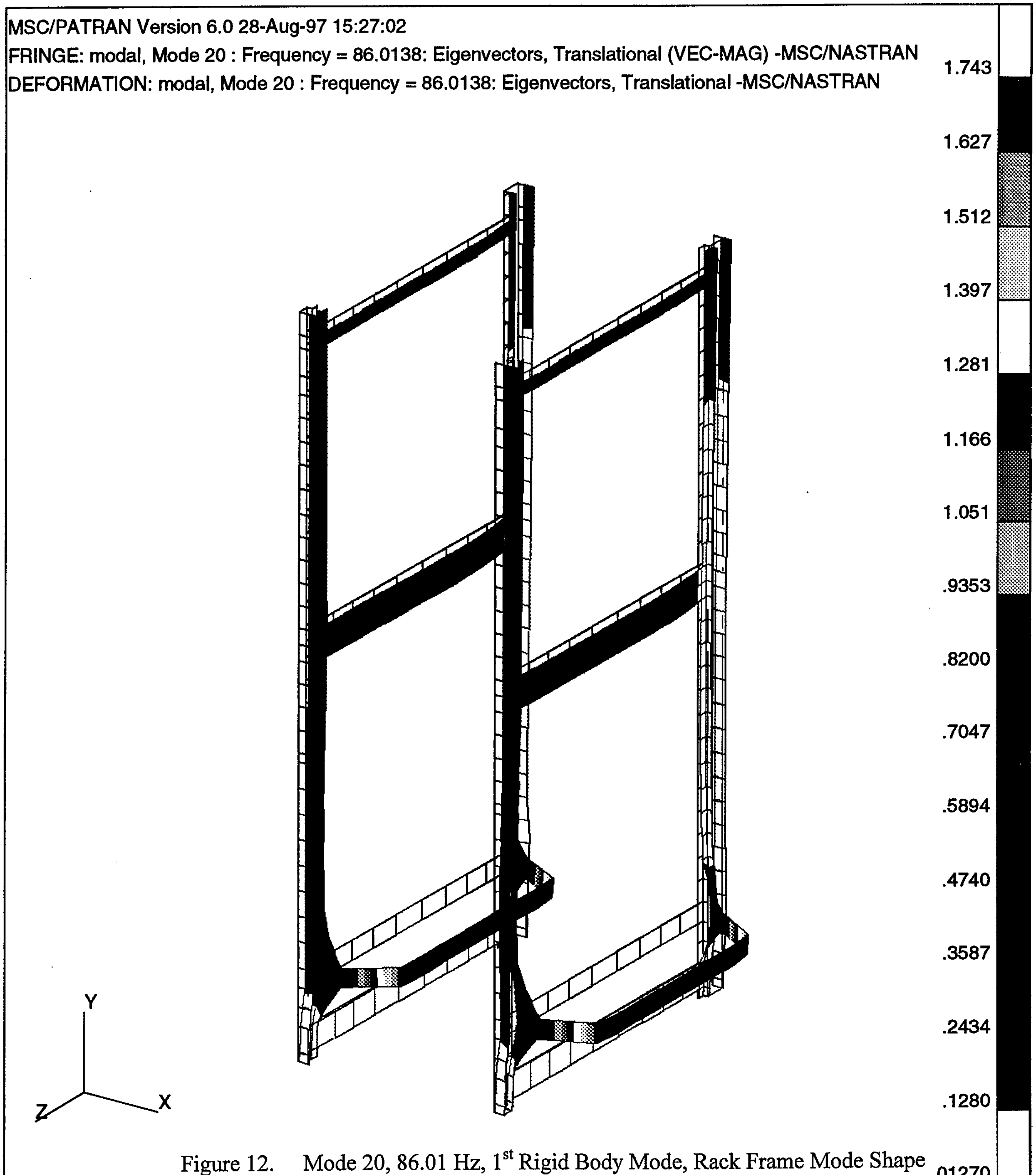

Figure 12. Mode 20, 86.01 Hz, 1st Rigid Body Mode, Rack Frame Mode Shape

describing the motions of the components which exhibit the greatest deformation value from the eigenvector. The deformations shown in the figures are not to scale with the parts.

Because there are 23 separate modes below 100 Hz, a general discussion follows proceeding up through the frequency range noting significant changes in the modes. Again, Appendix A gives a complete listing of the characteristics of all the modes below 100 Hz.

Because the structure is composed of many separate sub-assemblies rigidly connected to each other, most of the modes can be characterized as local bending modes. This includes but is not limited to n^{th} order beam and plate bending modes. Additionally there are what can be called rigid-body modes. These are characterized as gross assembly motion to include the entire cabinet structure and individual sub-assembly motion such as fore and aft motion of a particular drawer.

The major participants in modes 1 and 2, 52.16 and 52.52 Hz respectively, are the cabinet side panels (Figures 7 and 8). This is because they are the largest continuous pieces in the cabinet and based on theory of vibrations would have the lowest natural frequency. The side panels are vibrating in a 1^{st} plate bending mode with each side in phase with the other. Mode 2 exhibits the same local side plate bending mode as well as the beginnings of local bending modes for the top cover plate and the CPU cover panel.

At approximately 57 Hz, the 2^{nd} side panel bending mode begins with an increase in the amplitude of the other panel bending modes. Mode 6 (Figure 9) at 58.35 Hz brings the advent of the rear panel 1^{st} plate bending mode. The next 13 modes are characterized by a progression through higher order plate bending modes for the side and rear panels along with a shift in which front panel has localized bending.

The first rigid body mode (mode 20) occurs at 86.01 Hz. This mode shape is characterized by an athwartships rocking motion (similar to the first mode of a fixed-free beam) with an exaggerated in phase motion of the CPU drawer. Figures 10 through 12 show these mode shapes for the overall structure, drawers, and rack frame respectively.

Modes 21 through 23 (up through 97.16 Hz) are a continuation of the higher order side, top, and rear panel bending modes. Mode 23, Figure 13, is characterized by the fifth

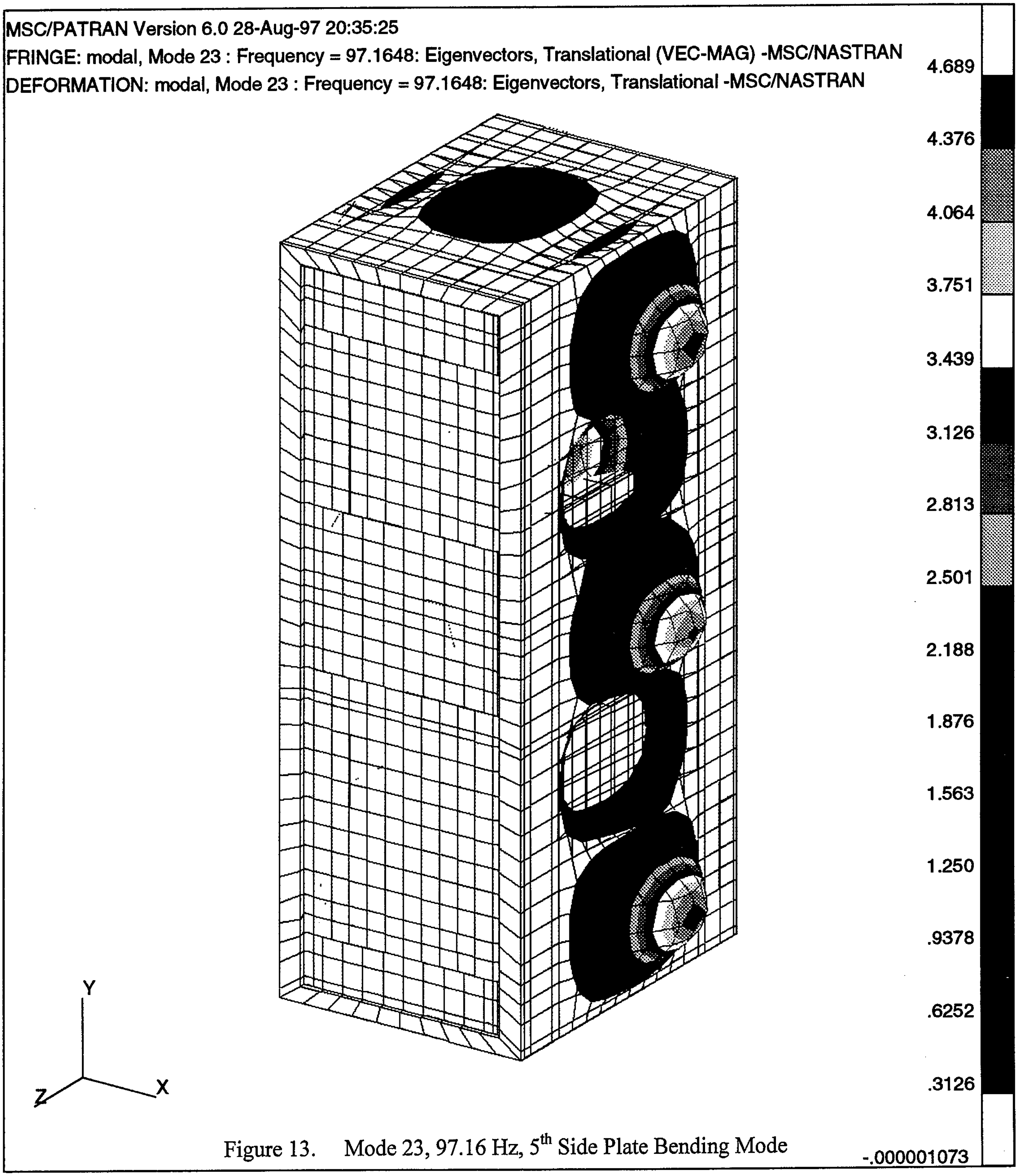

Figure 13. Mode 23, 97.16 Hz, 5th Side Plate Bending Mode

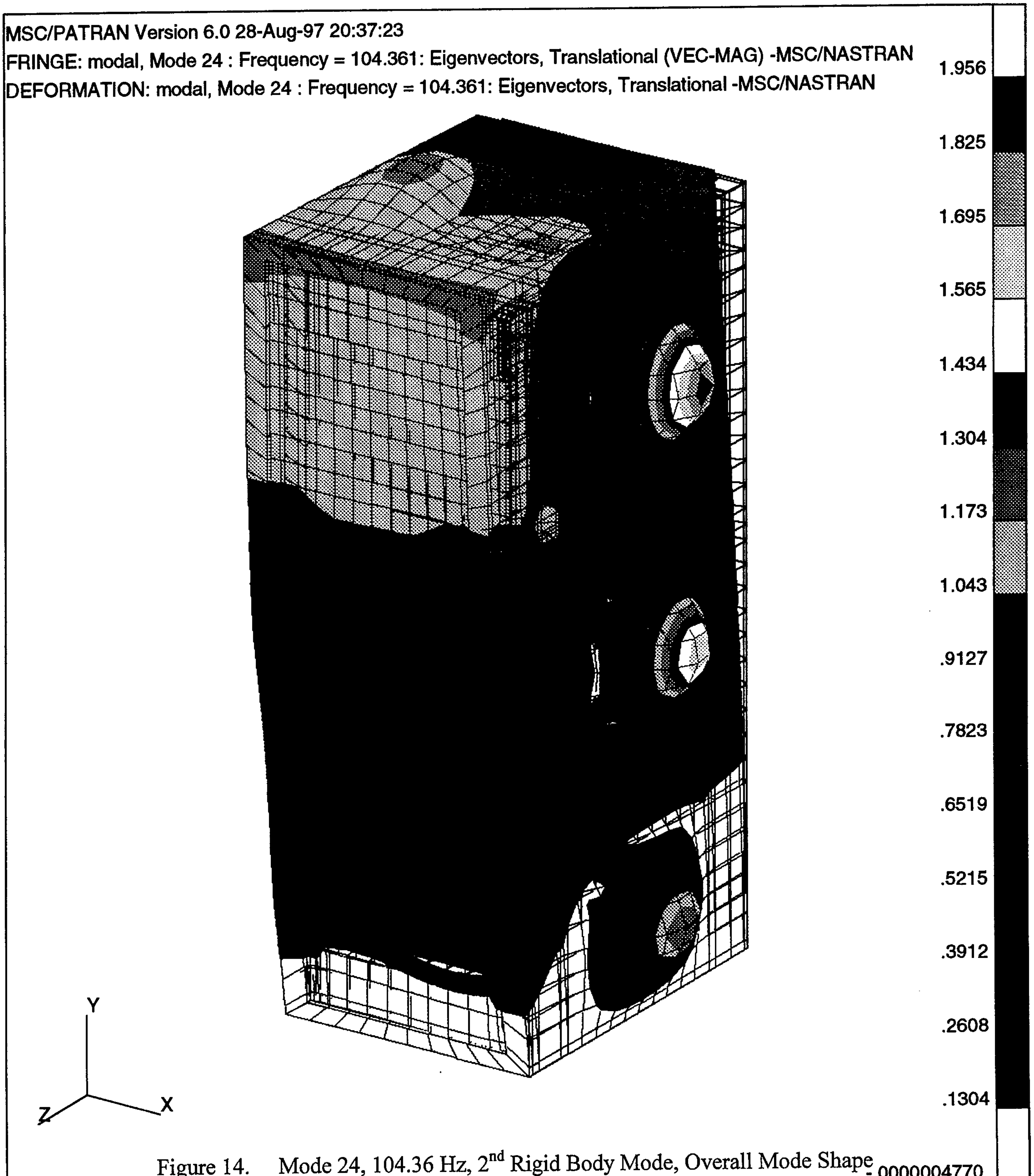

Figure 14. Mode 24, 104.36 Hz, 2nd Rigid Body Mode, Overall Mode Shape

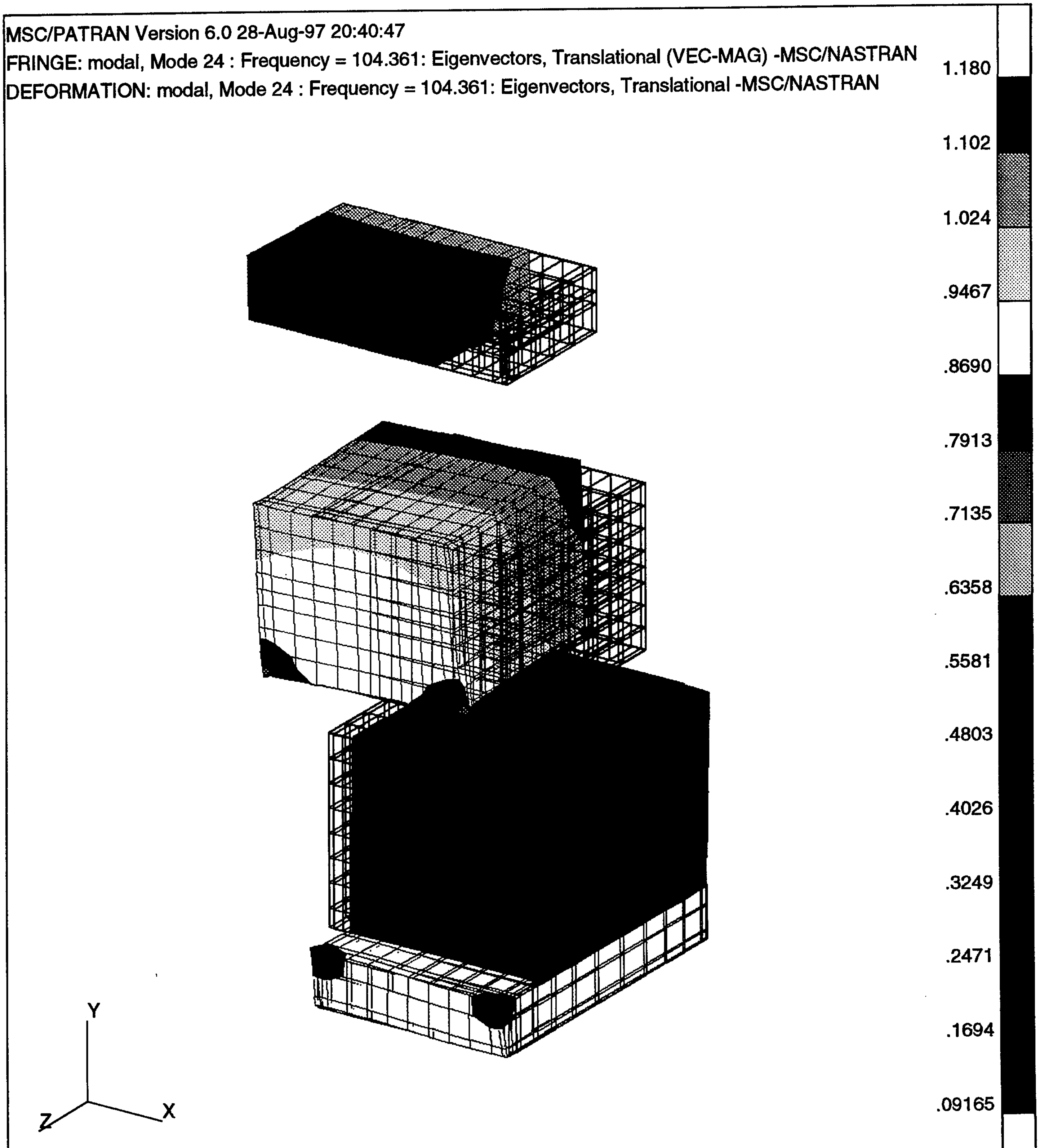

Figure 15. Mode 24, 104.36 Hz, 2nd Rigid Body Mode, Drawer Mode Shape

15

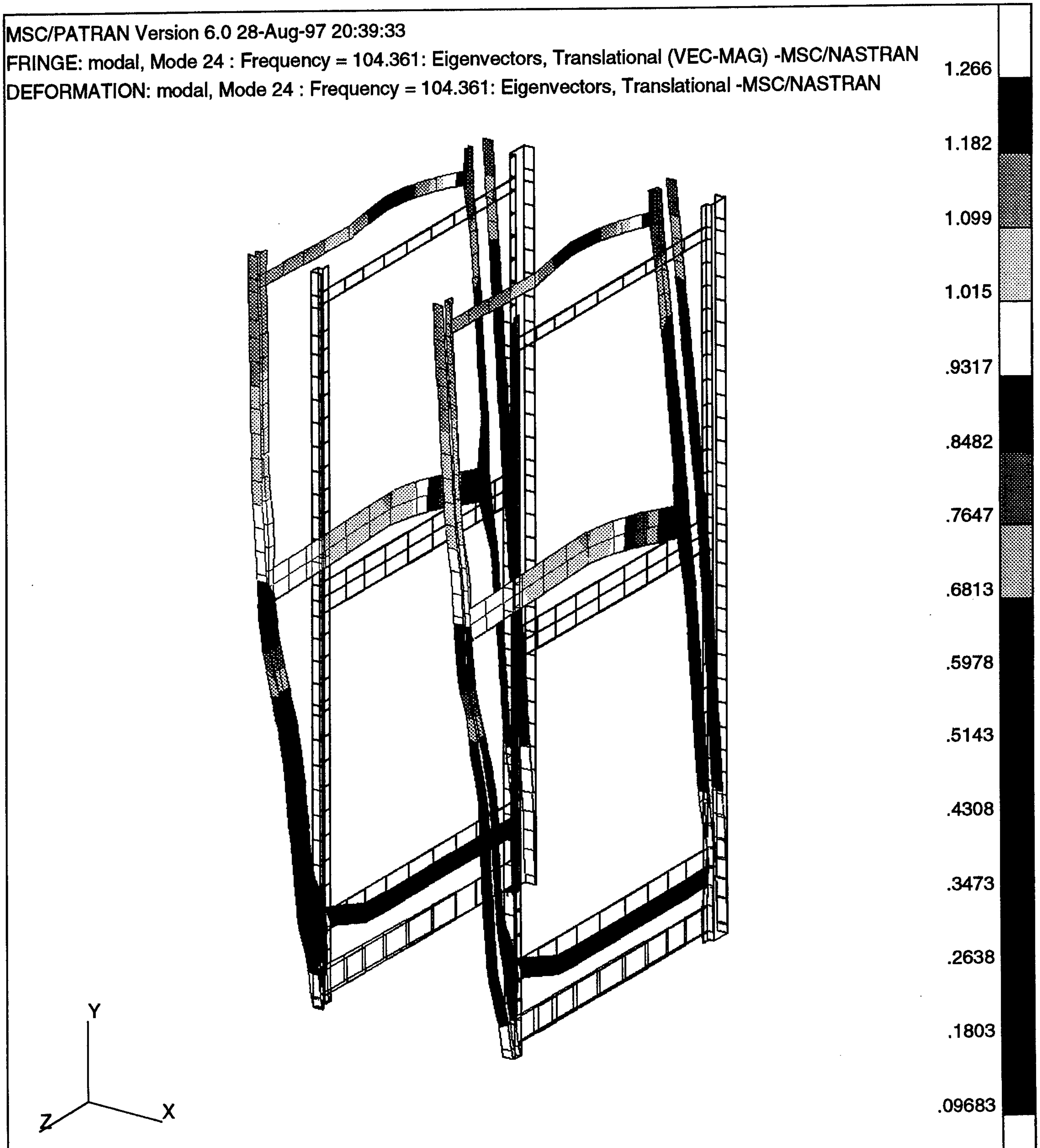

Figure 16. Mode 24, 104.36 Hz, 2nd Rigid Body Mode, Rack Frame Mode Shape

16

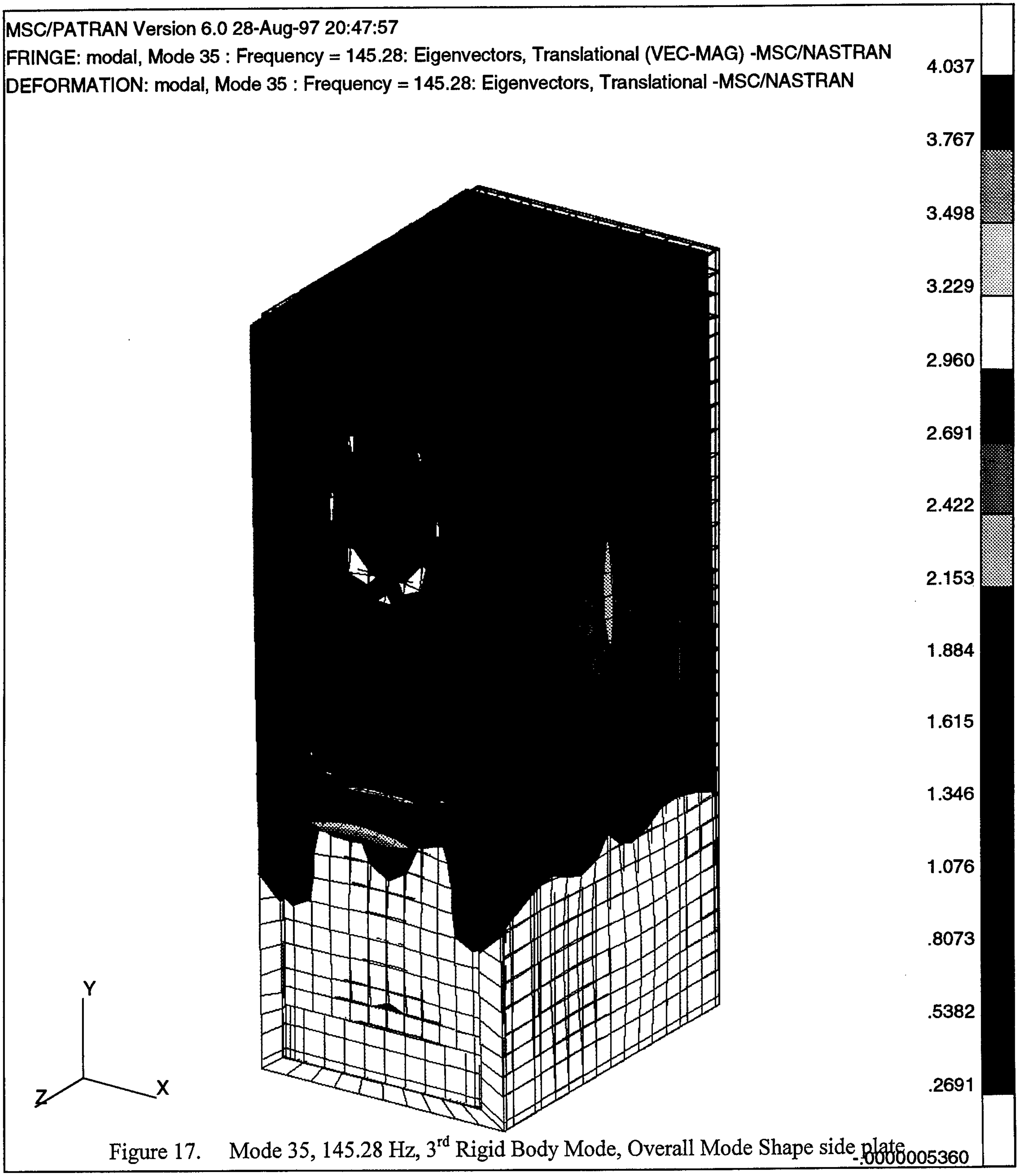

Figure 17. Mode 35, 145.28 Hz, 3rd Rigid Body Mode, Overall Mode Shape side plate

bending mode with each side in phase and a first order top plate bending mode influenced by the side plate motion.

In addition to the modes below 100Hz, mode 24 at 104.36 Hz is the second rigid-body mode. This mode is characterized by an athwartship first bending mode with a small twisting moment about the y-axis and some local plate bending modes. Additionally, the drawers exhibit an overall second bending moment. Figures 14 through 16 show the motion of the overall cabinet, the rack frame and the drawers, respectively. Finally, mode 35 at 145.28 Hz (Figure 17) exhibits the third rigid-body mode, a fore-aft first bending moment with some local plate bending modes.

6. DISCUSSIONS

Because the TAC-4 family of ruggedized racks is designed to allow the use of non-militarized equipment in a naval military environment on a variety of platforms, it is imperative that the rack perform as required by all shock and vibration specifications. Because actual shock testing of every rack with each of its varying configurations is prohibitively expensive and time consuming, a computerized approach would be faster and much more cost effective. A computerized approach to shock testing would also allow for a rapid tool for the designer to see how a change in the design affects the response of the model.

The goal of this work was to construct a finite element model and perform a model analysis of the 72 Inch TAC-4 ruggedized rack for CLIN 0003AA. This is only the first step. The preceding analyses must be performed on the rack system with the isolation mounts included in the model. Once this is complete, a transient analysis of this model will be analyzed using a base excitation. For this transient input, a half sine wave acceleration with a maximum amplitude and duration simulating a ship shock detonation can be used. Also, actual base velocity transients can be used from the test results of the navy medium weight shock machine and barge testing. The results of these computer simulations can then be compared to the actual shock test data.

Next a sensitivity analysis of the model will be performed by changing one property at a time in order to better understand the interrelation of the rack components. First, damping sensitivity will be investigated to determine the extent of the effect of structural damping on the system. These results can then be used to determine the actual structural damping of the rack system. Next, dimensional sensitivity must be performed by changing the dimensions of certain parts of the model, most notably the individual component thicknesses. This will determine what effect these different components have on the overall vibration characteristics of the rack system.

Finally, an comparison of the results from the transient analysis will be compared to the results of the approximating method used by the Dynamic Design Analysis Method (DDAM) which is another standard used by NAVSEA for determining the shock response of a shipboard component. This will provide a standard by which to judge the finite element method to a long-standing accepted standard.

Computerized modeling provides a cost-effective and time-efficient alternative to actual component testing. This frees defense money and ultimately improves the quality of the product, allowing for naval tactical computer technology to keep pace with the technological developments of commercial industry.

APPENDIX A: MODE SHAPES

The origin is located at the left, rear, bottom corner of the rack system. The x-axis is the athwartships (transverse) direction, the y-axis is the vertical direction, and the z-axis is the fore-aft (longitudinal) direction. The "MAX" parameter used in the table is not a physical dimensional value, but a relative quantity and not scaleable. The magnitude of this value shows which component's vibrations dominates in that particular mode.

MODE 1 FREQUENCY 52.16 Hz

PART	MAX	DESCRIPTION
Side Panels	4.445	1st plate bending mode, in phase

MODE 2 FREQUENCY 52.52 Hz

PART	MAX	DESCRIPTION
Side Panels	4.795	1st plate bending mode, in phase
Top Panel	.6393	1st plate bending mode, in phase
CPU Cover Panel	.6393	Cupping along top edge

MODE 3 FREQUENCY 57.17 Hz

PART	MAX	DESCRIPTION
Side Panels	2.754	2nd plate bending mode, in phase
Top Panel	1.102	1st plate bending mode
CPU Cover Panel	8.262	1st plate bending mode
Front Panel Filler Plates	4.957	1st plate bending mode

MODE 4 FREQUENCY 57.35 Hz

PART	MAX	DESCRIPTION
Side Panels	4.922	2^{nd} plate bending mode, in phase
Top Panel	.6563	2^{nd} plate bending mode

MODE 5 FREQUENCY 57.39 Hz

PART	MAX	DESCRIPTION
Side Panels	3.906	2^{nd} plate bending mode
Top Panel	1.420	1^{st} plate bending mode
CPU Cover Panel	3.551	1^{st} plate bending mode
Front Panel Filler Plates	3.551	1^{st} plate bending mode

MODE 6 FREQUENCY 58.35 Hz

PART	MAX	DESCRIPTION
Front Panel Filler Plates	7.515	1^{st} plate bending mode
CPU Cover Panel	2.004	1^{st} plate bending mode
Back Panel	5.010	1^{st} plate bending mode

MODE 7 FREQUENCY 58.75 Hz

PART	MAX	DESCRIPTION
Front Panel Above Monitor	7.336	1^{st} plate bending mode
Back Panel	5.379	1^{st} plate bending mode

MODE 8 FREQUENCY 60.30 Hz

PART	MAX	DESCRIPTION
Front Panel Filler Plates	17.52	1st plate bending mode
CPU Cover Panel	5.841	Cupping along lower edge

MODE 9 FREQUENCY 64.09 Hz

PART	MAX	DESCRIPTION
Side Panels	1.367	3rd plate bending mode, in phase
Back Panel	6.835	2nd plate bending mode
Top Panel	2.734	1st plate bending mode

MODE 10 FREQUENCY 64.23 Hz

PART	MAX	DESCRIPTION
Side Panels	3.673	3rd plate bending mode, out of phase
Back Panel	2.755	2nd plate bending mode
Top Panel	6.886	1st plate bending mode

MODE 11 FREQUENCY 65.12 Hz

PART	MAX	DESCRIPTION
CPU Cover Panel	6.086	Cupping at upper edge
Front Panel Filler Strip, lwr	30.43	1st Plate bending mode

MODE 12   FREQUENCY 65.82 Hz

PART	MAX	DESCRIPTION
Side Panels	4.740	3rd plate bending mode, out of phase
Top Panel	.9479	2nd plate bending mode

MODE 13   FREQUENCY 66.67 Hz

PART	MAX	DESCRIPTION
Rear Isolation Mount Plate	13.18	1st plate bending mode

MODE 14   FREQUENCY 68.19 Hz

PART	MAX	DESCRIPTION
Side Panels		3rd plate bending mode
Top Panel	8.077	1st plate bending mode

MODE 15   FREQUENCY 72.44 Hz

PART	MAX	DESCRIPTION
CPU Cover Panel	15.49	2nd plate bending mode
Front Panel Filler Plates	8.263	1st plate bending mode

MODE 16   FREQUENCY 73.72 Hz

PART	MAX	DESCRIPTION
Rear Panel	7.106	3rd plate bending mode

MODE 17 FREQUENCY 76.43 Hz

PART	MAX	DESCRIPTION
Filler Plate Above Monitor	15.77	2nd Plate bending mode

MODE 18 FREQUENCY 78.49 Hz

PART	MAX	DESCRIPTION
Side Panels	4.703	4th plate bending mode, in phase
Top Panel	1.568	2nd plate bending mode

MODE 19 FREQUENCY 79.63 Hz

PART	MAX	DESCRIPTION
Side Panels	4.652	4th plate bending mode, in phase
Top Panel	2.481	1st plate bending mode

MODE 20 FREQUENCY 86.01 Hz

PART	MAX	DESCRIPTION
Overall Cabinet	.3745	Athwartships, 1st bending mode
Side Panels	.6242	Asymmetrical 3rd Bending Mode, in phase
CPU	1.873	large athwartships displacement
Monitor/PDU	.3917	small athwartships displacement
Rack Frame	1.743	corresponds with drawers

MODE 21 FREQUENCY 88.43 Hz

PART	MAX	DESCRIPTION
Rear Panel	7.094	4th plate bending mode

MODE 22 FREQUENCY 95.23 Hz

PART	MAX	DESCRIPTION
Side Panels	4.502	5th plate bending mode, out of phase
Top Panel	2.701	2nd plate bending mode

MODE 23 FREQUENCY 97.16 Hz

PART	MAX	DESCRIPTION
Side Panels	4.689	5th plate bending mode, out of phase
Top Panel	1.876	1st plate bending mode, with side influence

LIST OF REFERENCES

1. Guide to Operations and Technical Information Manual for Standard Configuration TAC-4 Rugged Racks, SAIC-Computer Systems, San Diego, Ca.

2. MSC/PATRAN Course Notes, MacNeal-Schwendler Corp., Los Angeles, Ca.

3. Joseph, J.A. (ed.), MSC/NASTRAN Users Manual, Vols. 1 & 2, MacNeal-Schwendler Corp., Los Angeles, Ca.

INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center — 2
   Cameron Station
   Alexandria, Virginia 22304-6145

2. Library, Code 52 — 2
   Naval Postgraduate School
   Monterey, California 93943-5002

3. Professor Y.S. Shin, Code ME/Sg — 1
   Department of Mechanical Engineering
   Naval Postgraduate School
   Monterey, California 93943

4. Lieutenant Mark H. Oesterreich, Code ME/Sg — 1
   Department of Mechanical Engineering
   Naval Postgraduate School
   Monterey, California 93943

5. Commander Dave Reidy, Code D4103 — 1
   NCCOSC, RDT&E Division
   TAC Project Office
   49184 Transmitter Road
   San Diego, CA 92152-7346

6. Larry Core, Code D4103 — 1
   NCCOSC, RDT&E Division
   TAC Project Office
   49184 Transmitter Road
   San Diego, CA 92152-7346

7. Russ Eyres, Code D4103 — 1
   NCCOSC, RDT&E Division
   TAC Project Office
   49184 Transmitter Road
   San Diego, CA 92152-7346

8. John Walker, Code D6625 — 1
   Commanding Officer, NCCOSC, RDT&E Div
   53560 Hull St
   San Diego, CA 92152-5001

9. LCDR Joseph M Iacovetta, Code D3201 1
Commanding Officer, NCCOSC, RDT&E Div
53560 Hull St
San Diego, CA 92152-5001

10. Adam Simonoff, Code B325 1
NSWC
Dahlgren, VA 2248-5000

11. Wynne Davis 1
c/o CHET
P.O. Box 280114
Naval Station Mayport, FL 32228

12. Mike Winette 1
NSWCCDSSMDUERD
1445 Crossways Blvd
Cheasapeake, VA 23320

13. Research Office 1
Code 09